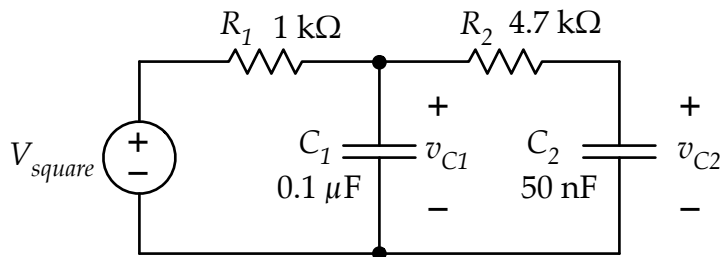


When there are multiple capacitors in a circuit and they cannot be reduced to a simple equivalent capacitance, the equation derived in class to describe the transient effects will not apply. (This is also true if there are multiple inductors in circuit.) In such cases, it is necessary to start from scratch and derive a *set* of differential equations to solve for the capacitor voltages. Generally, this can be quite tedious, and so we will often resort to SPICE to calculate the transients.

For the circuit below, use the node-voltage method to derive a coupled set of equations that *could* be used to find v_{C1} and v_{C2} . *You do not need to solve these.* (Note that your equations will have terms that include the time derivatives of the capacitor voltages.)

Then use PSPICE to simulate the transient behavior of the capacitor voltages in the circuit. For the source, use a square-wave with 0 V as the minimum, 10 V as the maximum and a frequency of 500 Hz – 1 ms high and 1 ms low.

You may want to work the the PSPICE transient tutorial that is on the class web site before trying this problem.



In submitting your solution to this homework include the set of (unsolved) equations, and the SPICE plots of v_{C1} and v_{C2} versus time.